Numerical Computation of Hypersonic Viscous Flow over a Sharp Leading Edge

J. C. TANNEHILL* AND R. A. MOHLING†
Iowa State University, Ames, Iowa
AND
J. V. RAKICH‡
NASA Ames Research Center, Moffett Field, Calif.

Nomenclature

C = Chapman-Rubesin constant, $\mu_w T_\infty/\mu_\infty T_w$
e = specific internal energy
E = total energy, $e+(u^2+v^2)/2$
k = coefficient of thermal conductivity
M = Mach number
p = pressure
Pr = Prandtl number
Re_x = Reynolds number
T = temperature
T_g = gas temperature at the wall
u, v = velocity components in x, y directions
$\bar{v}$ = rarefaction parameter, $M_\infty(C/Re_x)^{0.5}$
x, y = distances along and normal to plate
γ = specific heat ratio
δ_{ij} = Kronecker delta
λ = mean free path of the gas
μ = viscosity coefficient
ρ = density
$\bar{\chi}$ = strong interaction parameter, $M_\infty{}^2\bar{v}$

Subscripts

t = total condition
w = wall value
∞ = freestream value

Theme

THE hypersonic rarefied flow near the sharp leading edge of a flat plate has been computed using a finite-difference solution of the complete, unsteady Navier-Stokes equations. The solution is advanced in time from the initial conditions until the steady-state solution is reached. The computational region extends from the freestream ahead of the plate to the strong-interaction regime. Both wall slip and temperature jump are included in the calculations. The numerical results compare favorably with experimental data and Monte Carlo simulations. In addition, it is shown for the first time that a numerical solution of the Navier-Stokes equations predicts that the wall pressure approaches the free-molecule limit at the leading edge.

Presented as Paper 73-200 at the AIAA 11th Aerospace Sciences Meeting, Washington, D.C., January 10–12, 1973; submitted January 19, 1973; synoptic received May 9, 1973; revision received October 3, 1973. Full paper available from AIAA Library, 750 Third Avenue, New York, N.Y. 10017. Price: Microfiche, $1.00; hard copy, $5.00. **Order must be accompanied by remittance.** This work was partially supported by NASA Ames Research Center under Grant NGR 16-002-038 and the Engineering Research Institute, Iowa State University, Ames, Iowa.

Index categories: Rarefied Flows; Supersonic and Hypersonic Flow.

* Assistant Professor, Department of Aerospace Engineering. Associate Member AIAA.

† Instructor, Department of Aerospace Engineering; now Staff Engineer at Martin Marietta Corp., Denver, Colo.

‡ Research Scientist. Associate Fellow AIAA.

Contents

During the past decade, a number of theoretical investigations have been conducted to compute the hypersonic leading edge flow. These investigations can be divided into two groups, depending on whether a kinetic theory approach or a continuum flow approach is used. Included in the first approach are the Monte Carlo simulation method and the method of solving the Boltzmann equation using the BGK approximation model.

In the second approach, most investigators have used simplified versions of the Navier-Stokes equations to obtain a solution. Several studies have assumed "local similarity" in both the viscous layer and shock wave, so that the Navier-Stokes equations can be approximated by ordinary differential equations. In addition, several investigators have used the thin layer approximation to derive a set of parabolic equations which can be solved using standard boundary-layer techniques. These equations have been used to solve the flow in the merged layer and strong-interaction regions. However, it becomes unrealistic to use these equations to compute the flow closer to the leading edge, because of the streamwise gradient terms that are neglected in their derivation. In fact, it becomes necessary to use at least the complete Navier-Stokes equations, in conjunction with slip boundary conditions, to compute the entire leading edge flowfield.

In the present study, the leading edge flow has been computed using a finite-difference solution of the complete, unsteady Navier-Stokes equations. Solutions of the complete Navier-Stokes equations for the sharp leading edge problem were made previously by Butler[1] using the fluid-in-cell and particle-in-cell methods, and by Cheng and Chen[2] using a finite-difference

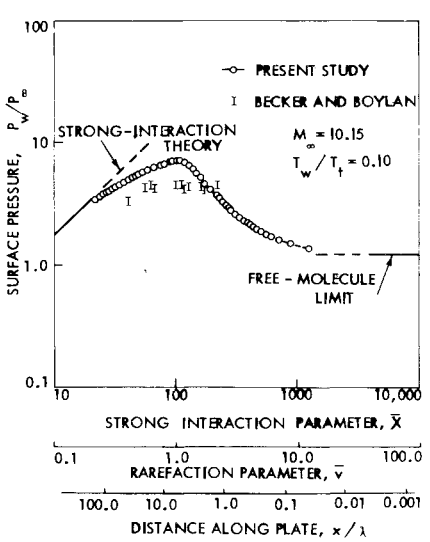

Fig. 1 Comparison of surface pressures.

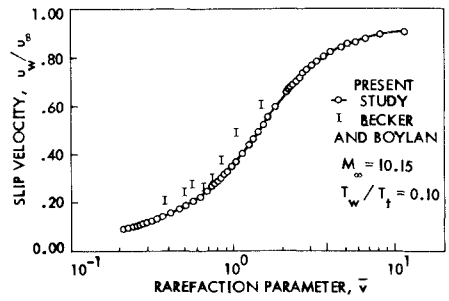

Fig. 2 Comparison of surface velocities.

treatment. However, in both studies the computational meshes employed were too coarse to adequately resolve the flowfield close to the leading edge. Without adequate resolution, the solution oscillates near the leading edge. On the other hand, if a fine grid is used throughout the computational region, it becomes prohibitively expensive to compute the entire leading edge flowfield.

These difficulties were avoided in the present study by patching together computational regions with different grid spacings, using a very fine grid near the leading edge. Thus, it was possible to compute the entire leading edge flowfield from the freestream to the strong interaction region. While it can be argued that the Navier-Stokes equations are not valid in the kinetic flow and transition regions, they do have a solution in these regions and it is believed that the solution is at least a first-order approximation to the actual flow for moderate Mach numbers. This is borne out by past experiments which have shown that the Navier-Stokes equations have a much greater range of validity than would be expected.

The Navier-Stokes equations can be written in the following vector form for a 2-D, unsteady flow:

$$(\partial \mathbf{U}/\partial t)+(\partial \mathbf{F}/\partial x)+(\partial \mathbf{G}/\partial y) = 0 \tag{1}$$

where

$$\mathbf{U} = \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ E \end{bmatrix}, \quad \mathbf{F} = \begin{bmatrix} \rho u \\ p+\rho u^2-\tau_{xx} \\ \rho uv-\tau_{xy} \\ Eu+pu+q_x-u\tau_{xx}-v\tau_{xy} \end{bmatrix},$$

$$\mathbf{G} = \begin{bmatrix} \rho v \\ \rho uv-\tau_{xy} \\ p+\rho v^2-\tau_{yy} \\ Ev+pv+q_y-u\tau_{xy}-v\tau_{yy} \end{bmatrix} \tag{2}$$

with the shear stress tensor and heat flux vector given by

$$\tau_{ij} = \mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)-\frac{2}{3}\mu\frac{\partial u_k}{\partial x_k}\delta_{ij}$$
$$q_j = -k\frac{\partial T}{\partial x_j} \tag{3}$$

In addition to these equations, the perfect gas equation of state has been used for all calculations.

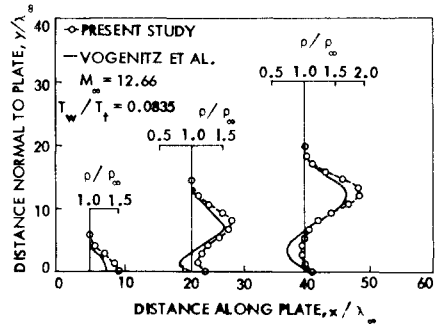

Fig. 3 Comparison of density profiles.

MacCormack's finite-difference scheme[3] was used to solve these equations at each interior grid point. This explicit scheme has second-order accuracy in both space and time. In addition, the flow variables at each boundary grid point must be specified to complete the solution. This was accomplished in the following manner. Along the inflow boundary of the first computational region, freestream conditions were maintained for all time. For each succeeding region, the inflow boundary conditions were determined from the outflow boundary conditions of the previous computational region. The flow variables along the top and outflow boundaries were determined using extrapolations of interior grid point data. At the wall (bottom boundary), the following slip conditions were imposed:

$$u_w = \left[\lambda\frac{\partial u}{\partial y}\right]_w$$
$$T_g = T_w+\left[1.996\left(\frac{\gamma}{\gamma+1}\right)\frac{\lambda}{P_r}\left(\frac{\partial T}{\partial y}\right)\right]_w \tag{4}$$

In addition, the normal velocity was set equal to zero, and the remaining flow variables (p_w, ρ_w) were determined by an extrapolation of interior data in conjunction with the equation of state. The solution in each region was advanced in time from the initial conditions until the steady-state solution was reached.

Typical results of this study are shown in Figs. 1–3. In Fig. 1, the computed surface pressures are compared with Becker and Boylan's experimental data for air.[4] The computed wall pressure begins with a value near the free-molecule limit, rises to a peak value exceeding the experimental data, and then approaches the pressure given by strong-interaction theory. This is believed to be the first numerical solution of continuum flow equations which has shown the decrease in wall pressure to the free-molecule value as the leading edge is approached. In addition, the computed pressure appears to asymptotically approach the strong-interaction theory near a rarefaction parameter ($\bar{v}$) of 0.20. This value agrees with the findings of many experimental investigations.

Figure 2 compares the computed slip velocity with the expermental results of Becker and Boylan. It is interesting to note that both the numerical and experimental results indicate a slip velocity of about 10% at the beginning of the strong-interaction region. And finally, Fig. 3 compares density profiles in the vicinity of the leading edge with the Monte Carlo solution of Vogenitz et al.[5] for the experimental flow conditions of Becker.[6] Reasonable agreement is achieved.

The complete paper presents additional examples which further collaborate the trends shown in Figs. 1–3. These results show that the Navier-Stokes equations, when solved by an accurate finite-difference method, yield realistic predictions of rarefied flows.

References

[1] Butler, T. D., "Numerical Solutions of Hypersonic Sharp-Leading-Edge Flows," *The Physics of Fluids*, Vol. 10, No. 6, June 1967, pp. 1205–1215.

[2] Cheng, S. I. and Chen, J. H., "A Finite-Difference Solution of Navier-Stokes Equations for Hypersonic Shock over a Sharp Leading Edge," presented at the Third International Conference on Numerical Methods in Fluid Dynamics, Paris, France, July 3–7, 1972.

[3] MacCormack, R. W., "Numerical Solution of the Interaction of a Shock Wave with a Laminar Boundary Layer," *Lecture Notes in Physics*, Springer-Verlag, New York, 1971, pp. 151–163.

[4] Becker, M. and Boylan, D. E., "Flow Field and Surface Pressure Measurements in the Fully Merged and Transition Flow Regimes on a Cooled Sharp Flat Plate," *Rarefied Gas Dynamics*, Suppl. 4, Vol. 2, edited by C. L. Brundin, Academic Press, New York, 1967, pp. 993–1014.

[5] Vogenitz, F. W., Broadwell, J. E., and Bird, G. A., "Leading Edge Flows by the Monte Carlo Direct Simulation Technique," *AIAA Journal*, Vol. 8, No. 3, March 1970, pp. 504–510.

[6] Becker, M., "Flat Plate Flow Field and Surface Measurements in Low Density Argon Between Continuum and Near Free Molecule Flow," *Rarefied Gas Dynamics*, Suppl. 5, Vol. 1, edited by L. Trilling and H. Wachman, Academic Press, New York, 1969, pp. 515–528.